



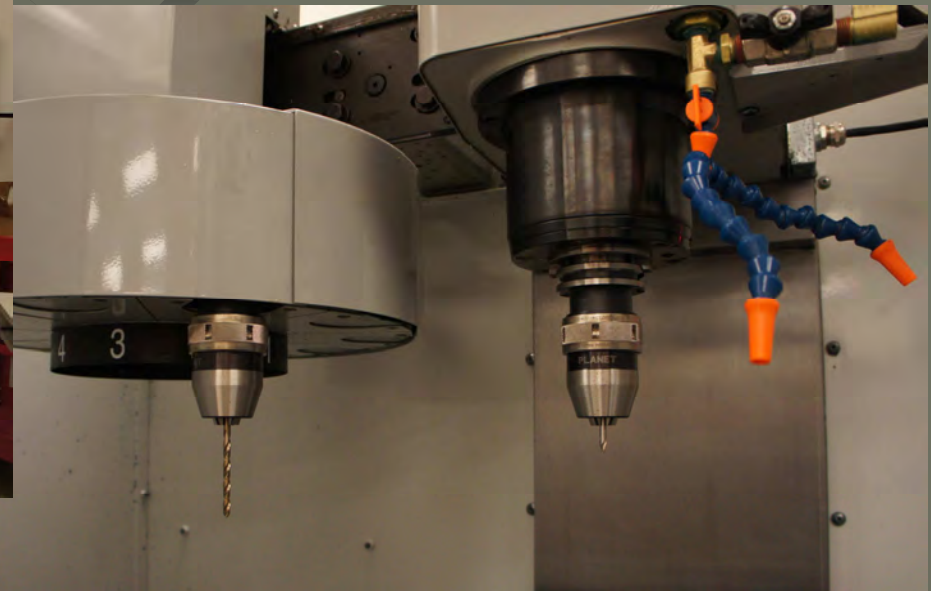
Introduction to CNC & CAM ENGI 7928

Getting started...

The CNC Lab

- ◆ Introduction to new software: Mastercam (2D milling)
- ◆ You will draw a model in SolidWorks - sketches must be fully defined
- ◆ You will use a tutorial handout to “machine” a part in Mastercam
- ◆ Tutorial: Mill_1_handout available in the 7928 folder on the S:\enr\courses\7928 folder
- ◆ When working in Mastercam- you must be diligent with the values you input, a typo could equal a crash.
- ◆ Mastercam Labs: Thurs., July 5, Fri. July 6 with TA available
- ◆ CNC Demonstration- in groups of 6 between July 11&12, 2-4pm
- ◆ NOTE :Mastercam is limited to 12 licenses only, therefore you will need to find time when a station is available for you to complete the lab

Memorial Engineering Student Machine Shop



- ◆ Haas Super Mini CNC Mill, 15 hp
- ◆ 10,000 rpm, 833 ipm feed cutting rate
- ◆ 10 Pocket umbrella tool changer
- ◆ Travel: X=16", Y=12", Z=10"

Memorial Engineering Student Machine Shop



- ◆ Mazak Quick Turn Nexus 100-II CNC Lathe, 15hp
- ◆ 6,000 rpm, 1180 ipm cutting feed rate
- ◆ 12 position tool changer
- ◆ Travel: X= 11", Z=12"

Many types of CNC machines & many outcomes!

- ◆ Lathes, Mills
- ◆ 2 to 5 Axis
- ◆ Lathe bar feeders
- ◆ Pallet Changer Centres



Photos compliments of Vincennes University www.vuhtec.org

Simple to complex

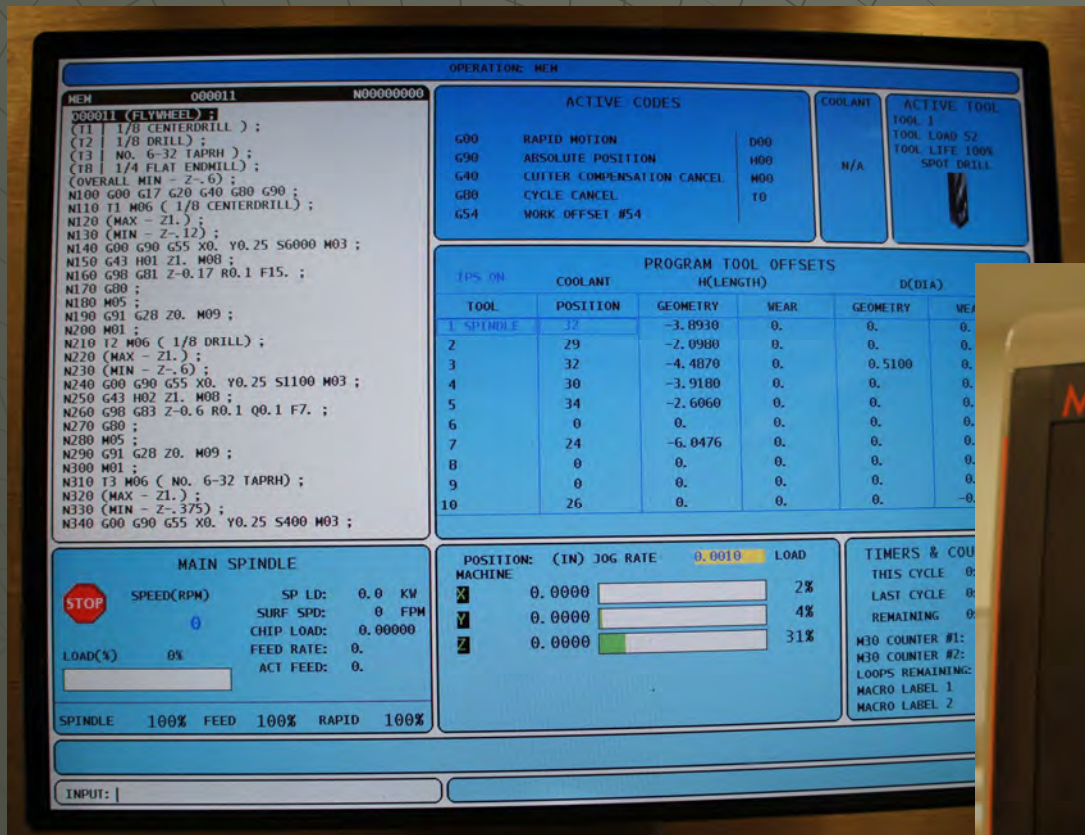


What is CNC?

- ◆ Used to automate machine tools to perform simple to complex machining operations
- ◆ Many controller languages are used i.e. conversational & G code
- ◆ Manually: code is used to build a program by manually entering commands using the key pad
- ◆ Software: for more complex or feature intense components we use CAM software to generate code for lengthy machining processes and /or complex surfaces

Writing Code

◆ G-Code



◆ Conversational



What is CAM?

- ◆ Computer Aided Manufacturing
- ◆ Used to assist manufacturers to generate CNC programs
- ◆ Not a simple “click of a button”
- ◆ CAM can be parameter heavy: tools must be selected, speeds, feeds, depth of cuts, efficient tool paths selected & verified, clearances checked, reduce program times etc.
- ◆ CAM software allows us to machine components **VIRTUALLY!**

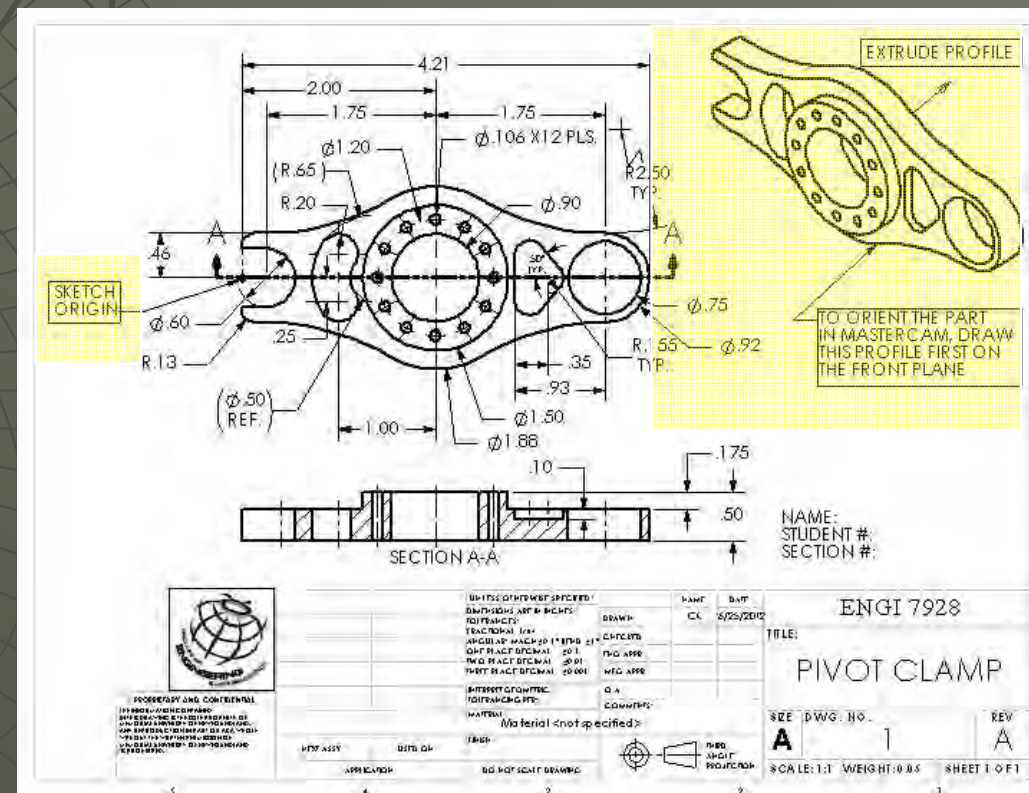
7928- Using Mastercam

- ◆ Introduction to Mastercam
- ◆ Importing SolidWorks models
- ◆ Selecting stock size
- ◆ Setting up machining parameters
- ◆ Selecting machining operations, tool types, feeds, speeds
- ◆ Determine axial & radial depth cuts
- ◆ Decide on rough & finish passes
- ◆ Verify toolpaths & cycle times
- ◆ Generate tool list, set up sheet & post operations

Design a solid model in SolidWorks:

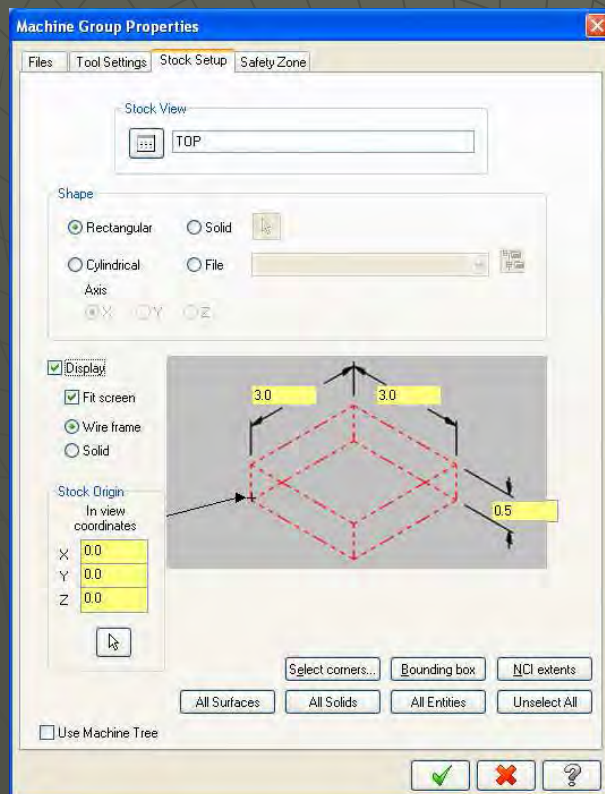
TIP: keep it simple

- ◆ Note: the front plane in SolidWorks will become the top plane in Mastercam.
- ◆ Use the recommended approach to drawing the solid model. In Mastercam we will make some minor modifications to reposition the origin.
- ◆ Save the SolidWorks model as an .x_t parasolid file
- ◆ Fully define all sketches in SW
- OR potential m/c
CRASH!



Mastercam is limited to 12 licenses in this lab

- It is available also available in your home room

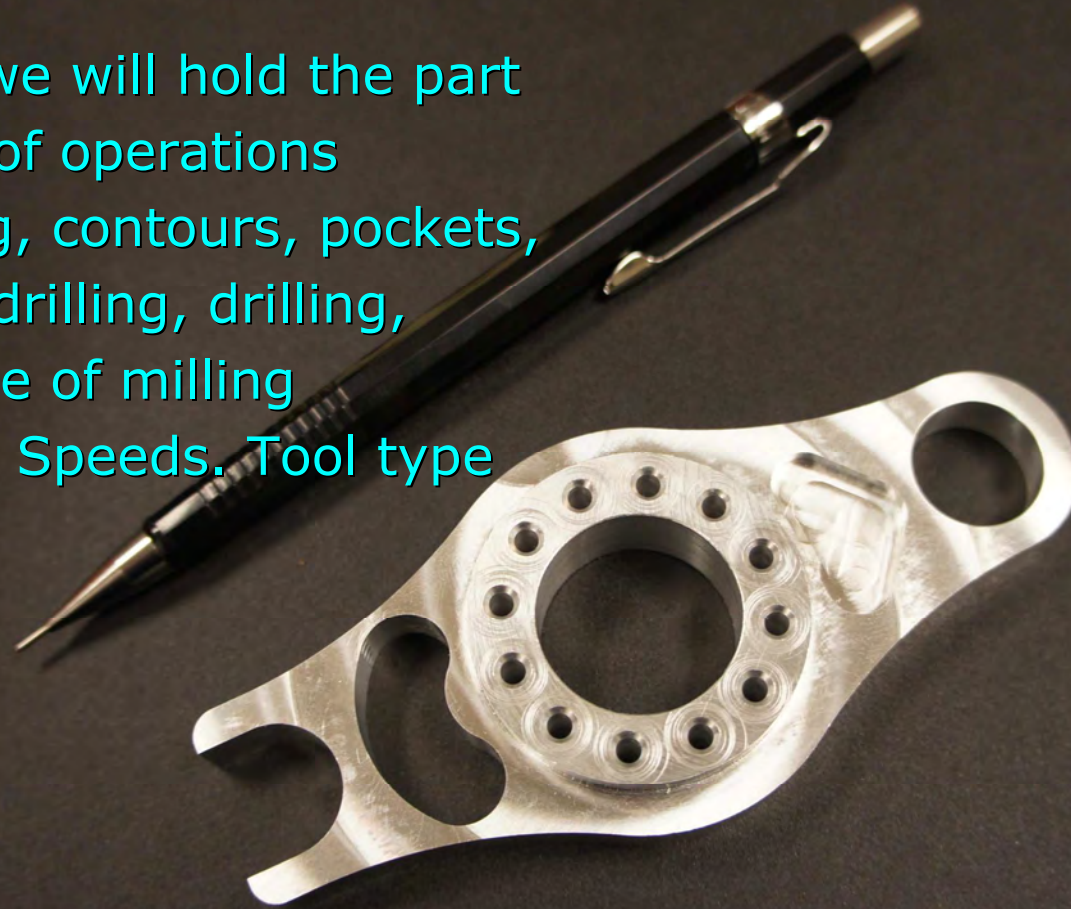


We will use *Mill_1_2012handout* in 7928 folder and follow follow the steps to set up stock size & stock origin

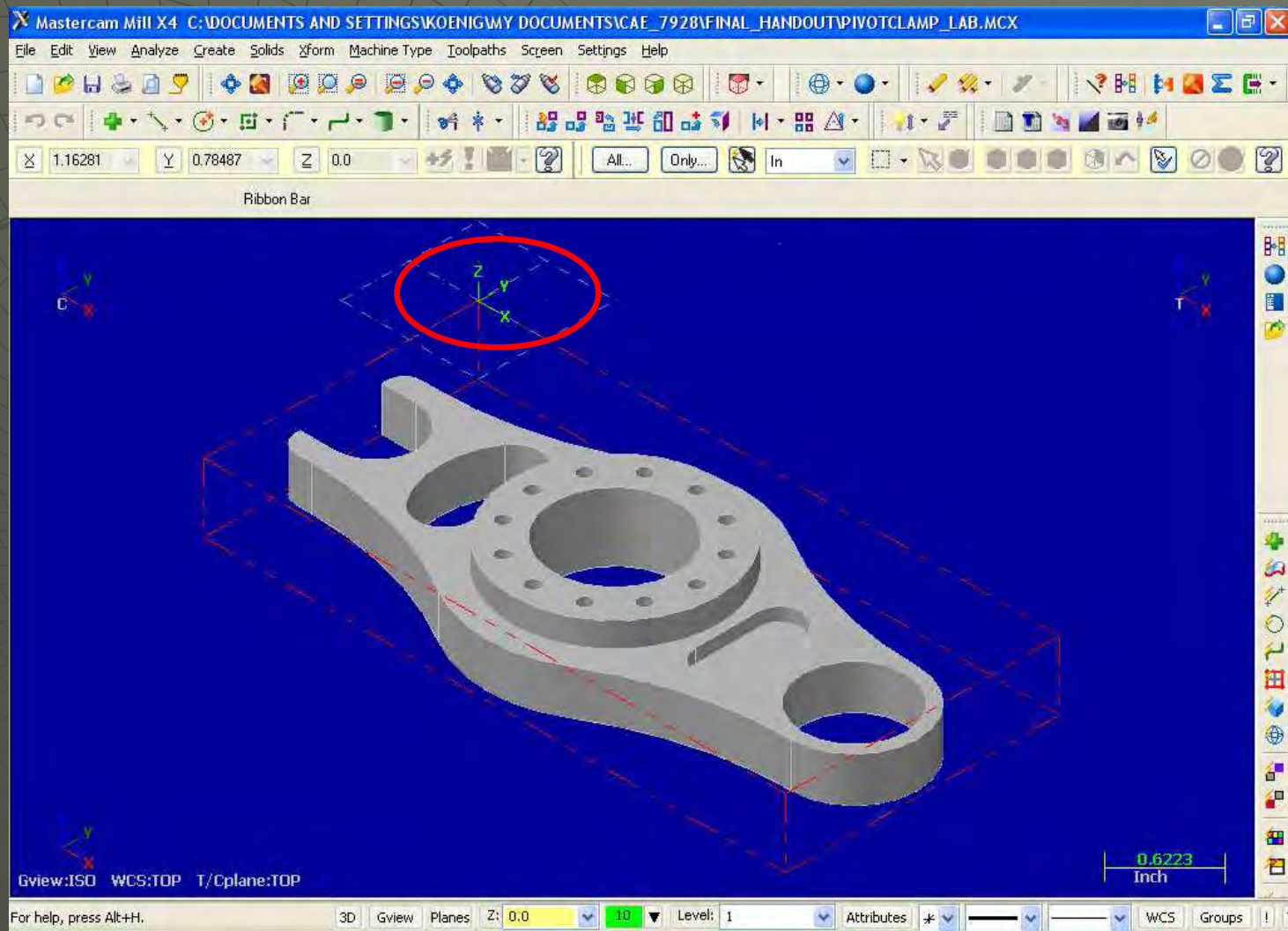
The project: Pivot Clamp

Consider:

- ◆ How we will hold the part
- ◆ Type of operations
- ◆ Facing, contours, pockets, spot drilling, drilling, & type of milling
- ◆ Feeds, Speeds. Tool type



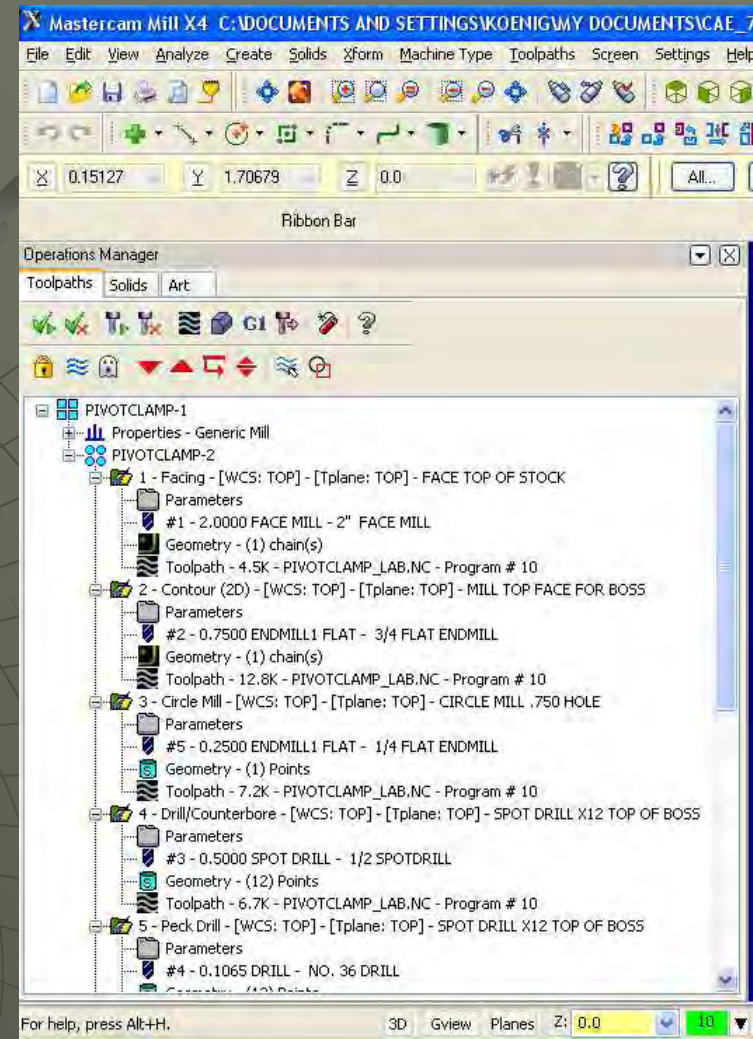
The Mill 1 handout provides the order & type of machining operations required, as well as tool selection & recommended feeds & speeds:
(note the location of the origin in Mastercam)



Machining steps:

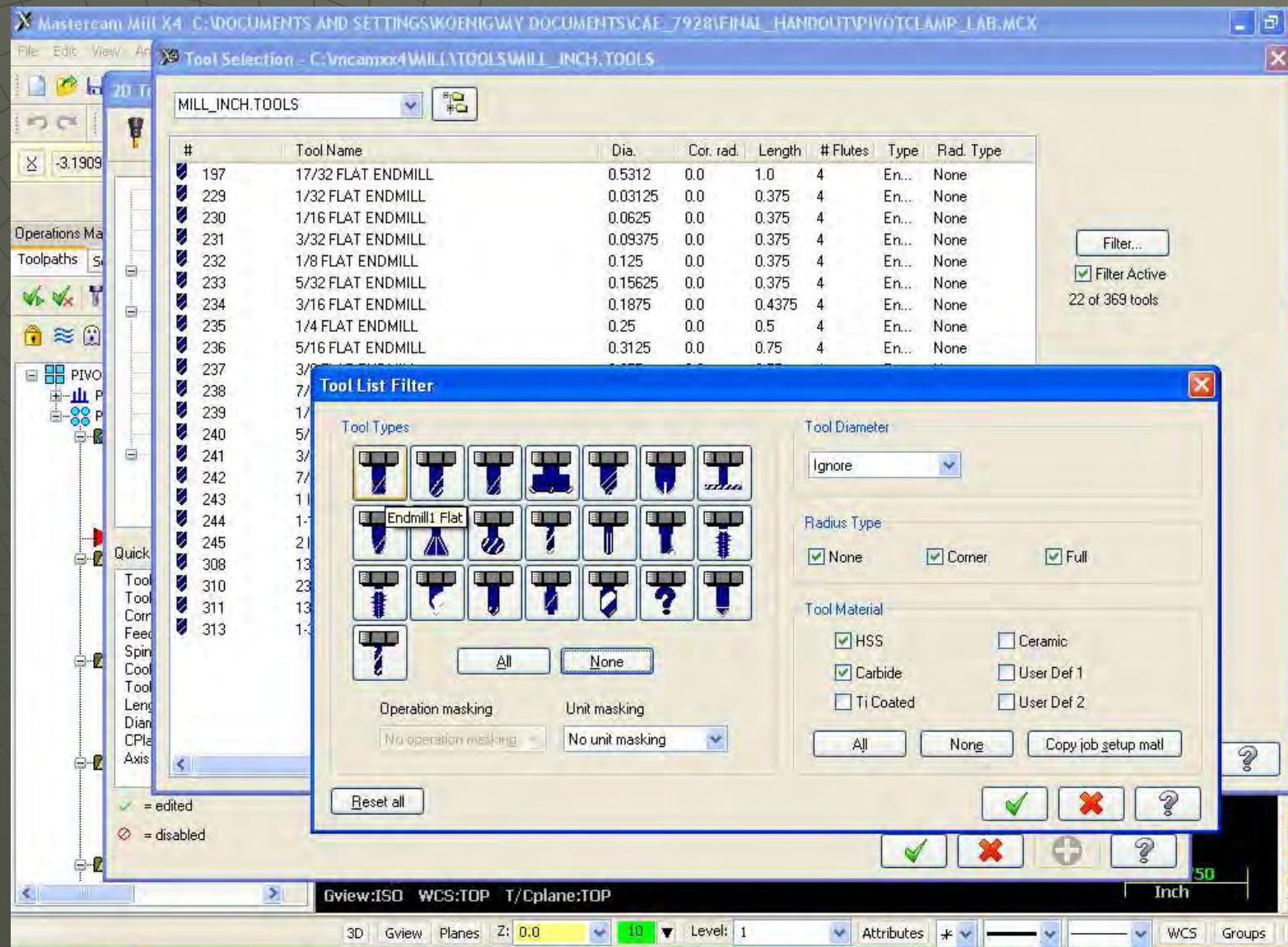
Tip: *Think about rough cuts & finish cuts*

In Mastercam the Operations Manager shows the machining operations, parameters, geometry & toolpath- we will create these operations.



Select tools from the library:

Tip: Filter, None, then select the tool required.



Enter parameters as show in handout.

Tip: Always enter a clear comment for each operation, this will appear in the code.

2D Toolpaths - Facing

Toolpath Type: **Tool**

Holder: []

Cut Parameters: []
Depth Cuts: []
Linking Parameters: []
Home / Ref. Points: []

Planes (WCS): []
Coolant: []
Canned Text: []
Misc Values: []
Axis Control: []
Axis Combination: []
Rotary Axis Control: []

Quick View Settings

Tool	2" Face Mill
Tool Diameter	2
Corner Radius	0
Feed Rate	10
Spindle Speed	1200
Coolant	On
Tool Length	2.1
Length Offset	1
Diameter Off...	1
CPlane / TP...	TOP
Axis Combin...	Default (1)

#	Tool Name	Dia.	Cor. ra
1	2" Face Mill	2.0	0.0
2	3/4 FLAT ENDMILL	0.75	0.0
3	1/2 SPOTDRILL	0.5	0.0
4	NO. 36 DRILL	0.1065	0.0
5	1/4 FLAT ENDMILL	0.25	0.0
9	1/2 FLAT ENDMILL	0.5	0.0

Right-click for options

Select library tool... ☐ Filter.Active

☐ To batch

Tool diameter: 2.0
Corner radius: 0.0
Tool name: 2" Face Mill
Tool number: 1 Length offset: 1
Head number: -1 Diameter offset: 1

Feed rate: 10.0 Spindle direction: CW
FPT: 0.0042 Spindle speed: 1200
Plunge rate: 5.0 SFM: 628.2723
☐ Force tool change ☒ Rapid Retract

Retract rate: 1.0696

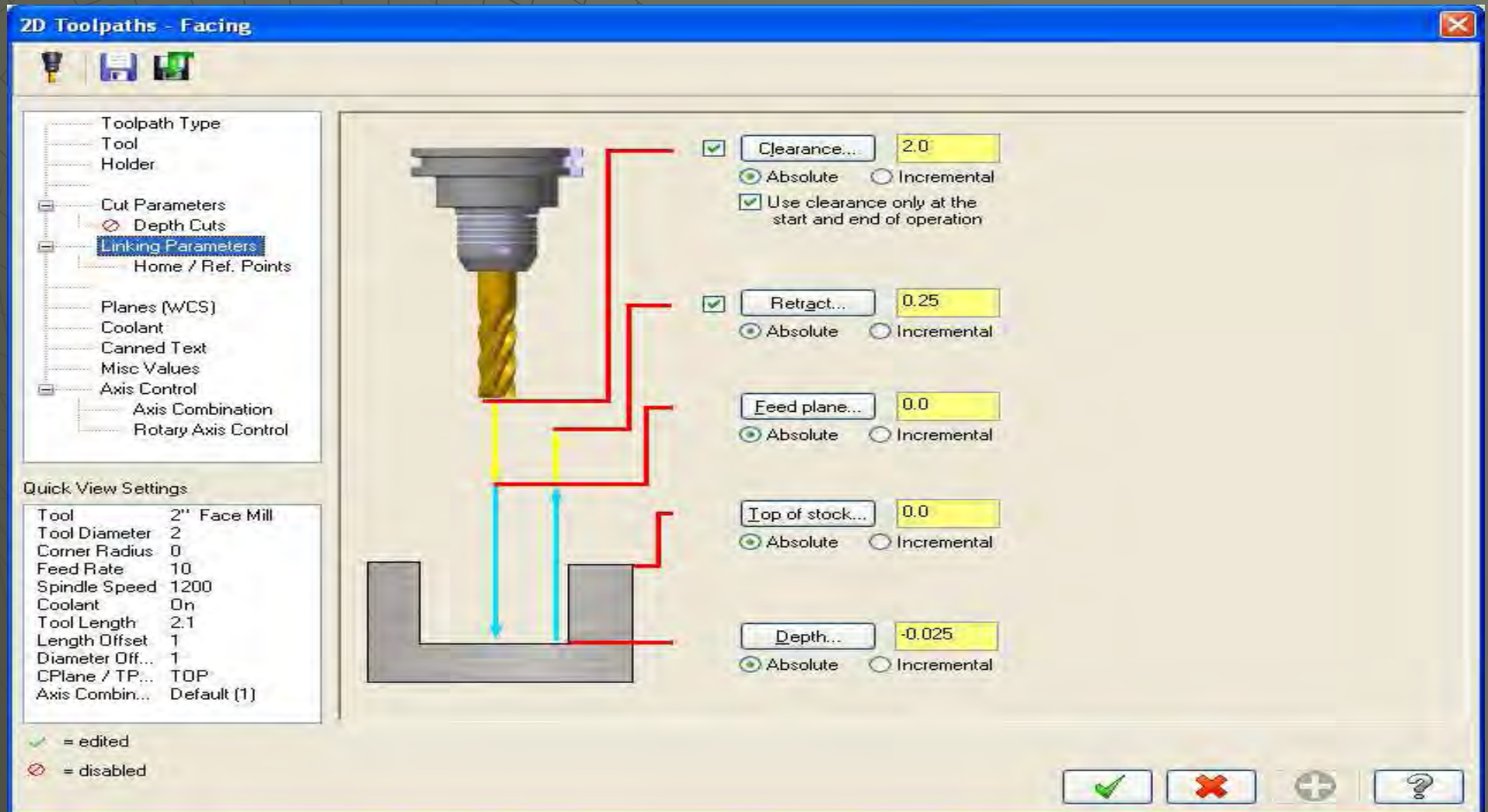
Comment: **FACE TOP OF STOCK**

Legend:
✓ = edited
✗ = disabled

Buttons: [] [] [] []

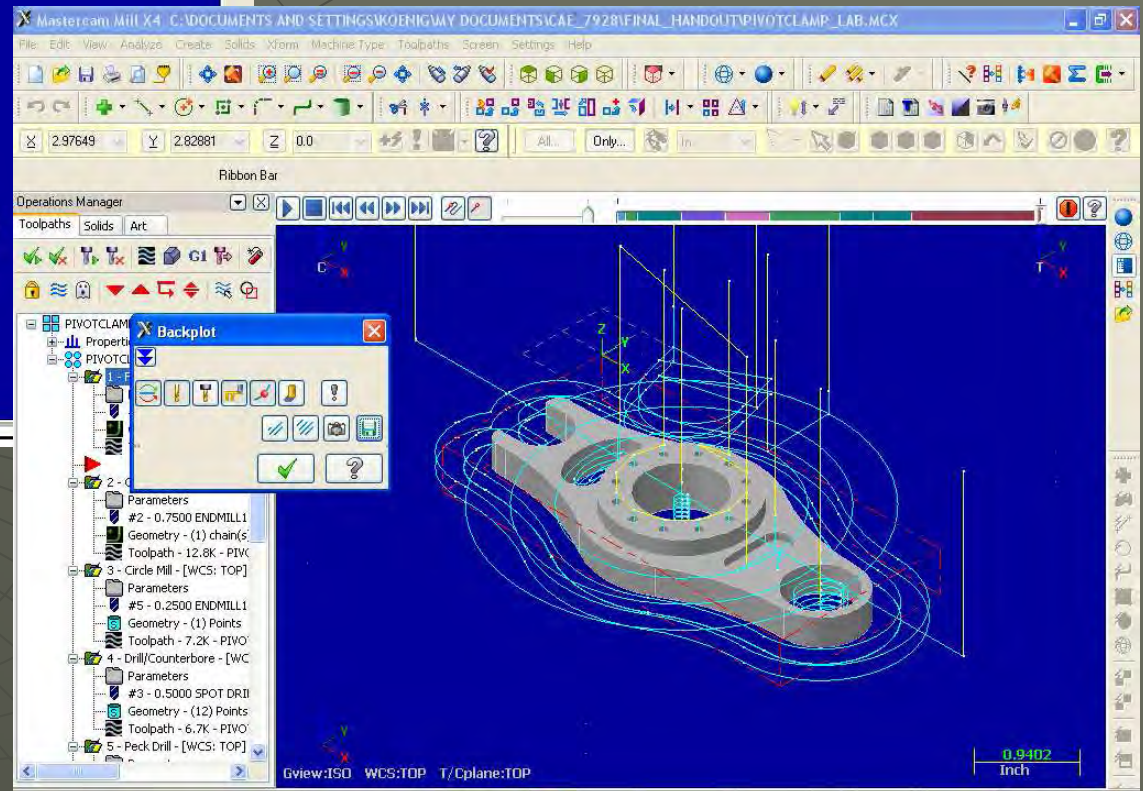
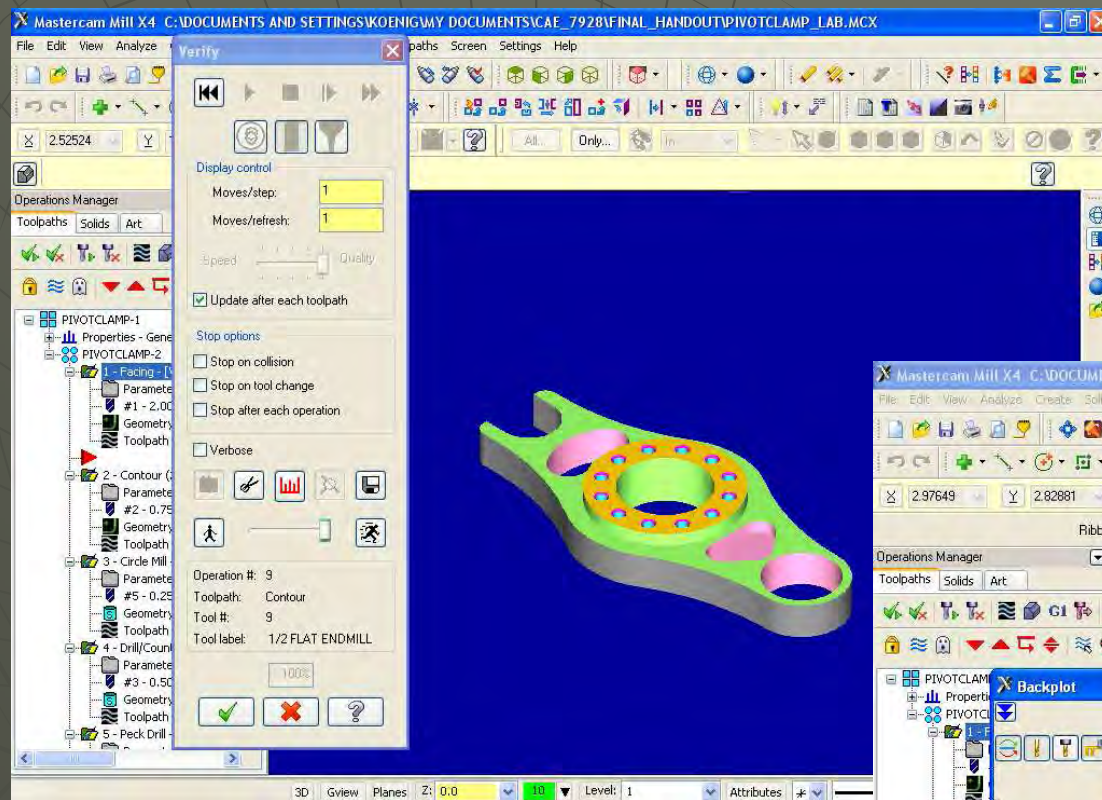
Enter the parameters for each machining operation in Mastercam

Tip: Be sure proper clearance is given along all axis around all work holding devices!

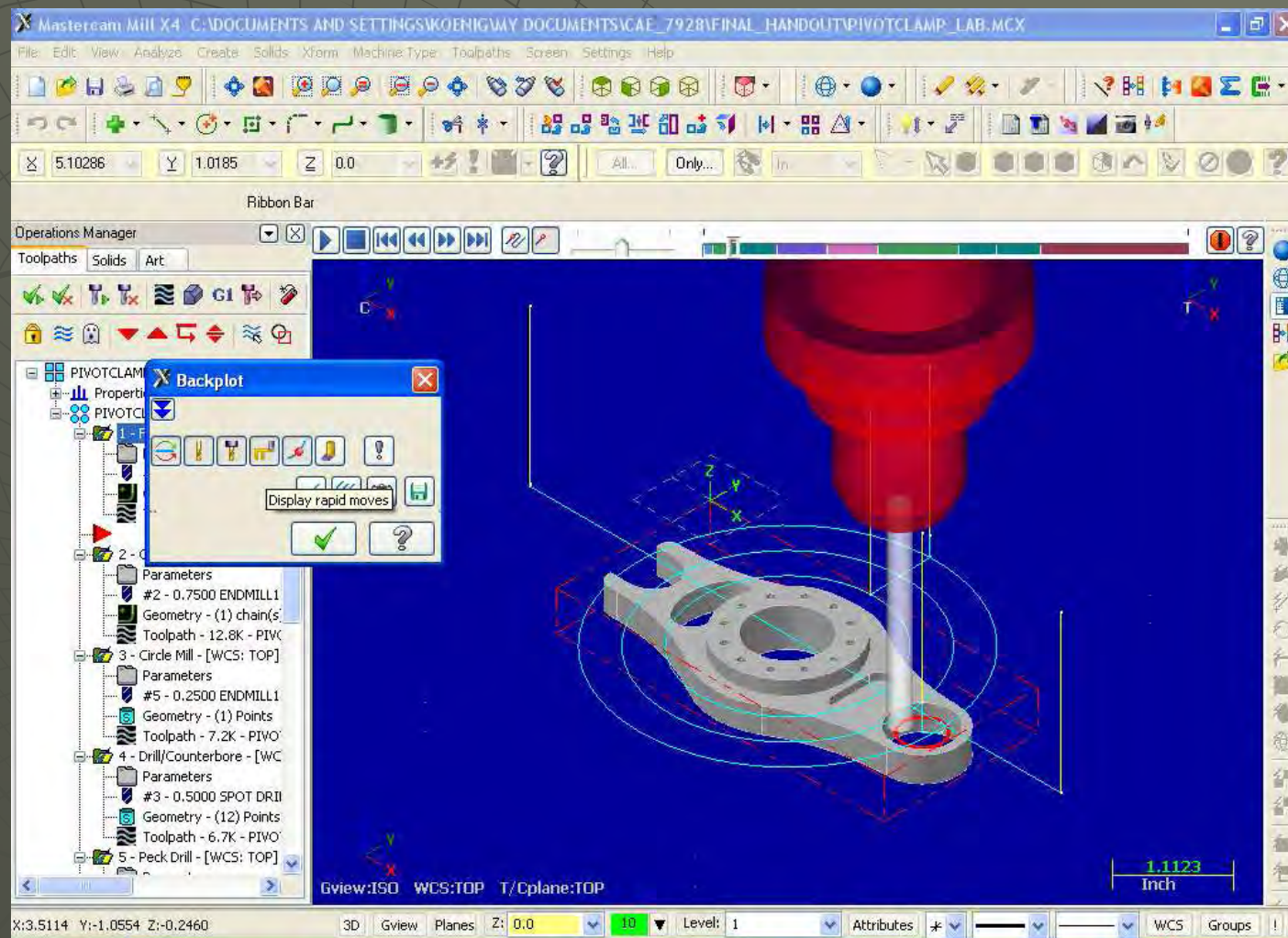


Use Verify & Backplot to check toolpaths, depth cuts, machining direction and more...

Tip: “Make your *virtual mistakes* in Mastercam”



Backplot will show the detailed machining steps created in each operation, tool, toolholder, rapids moves & more.


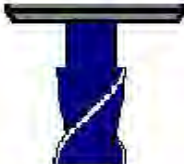




Generate a detailed Tool List

Tool List of PIVOTCLAMP_LAB.MCX

File Fonts Document Help

Proj./Part No.: 7928 LAB MILL II 2012 Date : 06/28/12
 Drawing No. : 1 Customer : FACULTY OF ENGINEERING
 Prog. No. : 10 Programmer : YOUR NAME

	Tool type :	2 Face mill 2" Face Mill			
	Manufact.code :				
	Chuck :				
	Tool Number :	1		Feedrate :	10
	Diameter :	2	RPM :	1200	Plunge feed r.:
	Corner radius :	0	Tip angle :	45	Diam. offset :
	Flute length :	1.575	Material :	ALUMINUM ...	Length offset :
	Overall length:	2.1	No flutes :	2	
	Tool type :	0.75 Endmill Flat 3/4 FLAT ENDMILL			
	Manufact.code :				
	Chuck :				
	Tool Number :	2		Feedrate :	10
	Diameter :	0.75	RPM :	1200	Plunge feed r.:
	Corner radius :	0	Tip angle :	0	Diam. offset :
	Flute length :	2	Material :	ALUMINUM ...	Length offset :
	Overall length:	3.75	No flutes :	4	
	Tool type :	0.5 Spot Drill 1/2 SPOTDRILL			
	Manufact.code :				
	Chuck :				
	Tool Number :	3		Feedrate :	10
	Diameter :	0.5	RPM :	2000	Plunge feed r.:
	Corner radius :	0	Tip angle :	118	Diam. offset :
	Flute length :	2	Material :	ALUMINUM ...	Length offset :
	Overall length:	3	No flutes :	2	
	Tool type :	0.1065 Drill NO. 36 DRILL			
	Manufact.code :				
	Chuck :				
	Tool Number :	4		Feedrate :	4
	Diameter :	0.1065	RPM :	4800	Plunge feed r.:

Ready NUM

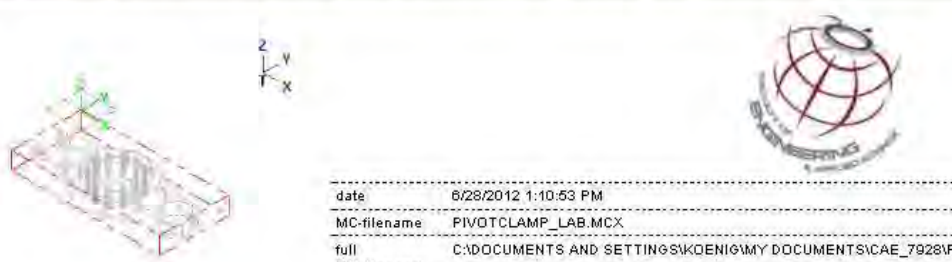
Generate a detailed Setup Sheet

Mozilla Firefox

File Edit View History Bookmarks Tools Help

file:///C:/DOCUMENT...INAL_HANDOUT/T.htm

file:///C:/DOCUMENTS AND SETTINGS/KOENIG/MY DOCUMENTS/CAE_7928/FINAL_HANDOUT/T.htm



date 6/28/2012 1:10:53 PM

MC-filename PIVOTCLAMP_LAB.MCX

full C:\DOCUMENTS AND SETTINGS\KOEING\MY DOCUMENTS\CAE_7928\FINAL_HANDOUT\PIVOTCLAMP_LAB.MCX

MC-filename






file description Original Import File name = C:\DOCUMENTS AND SETTINGS\KOEING\MY DOCUMENTS\CAE_7928\FINAL_HANDOUT\PIVOTCLAMP.X_T

Machine group PIVOTCLAMP-1

Group comment

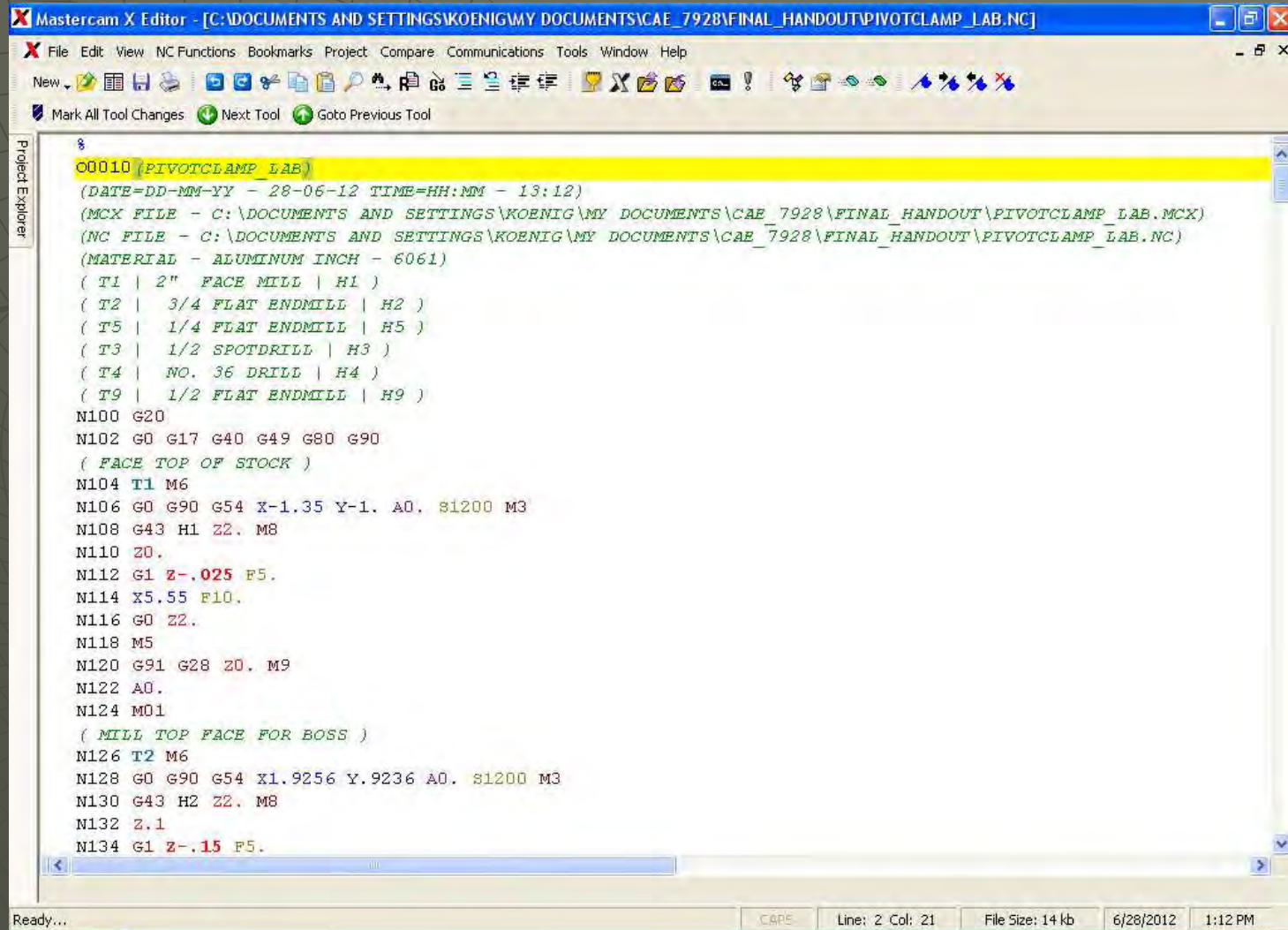
Material ALUMINUM inch - 6061

Nullpunkt

tl_type	tl_name	tl_nr	comment	op_type	stock	z_min	cycle time	coolant	c-comp	tl_dia_offs	rpm	top of stock
	Face Mill 2" Face Mill	1	FACE TOP OF STOCK	Facing	0	-0.03	00:00:42	Flood		1	1200	abs 0
	End Mill 3/4 FLAT ENDMILL	2	MILL TOP FACE FOR BOSS	Contour	0	-0.16	00:03:00	Flood	PC, left	2	1200	abs -0.03
	End Mill 1/4 FLAT ENDMILL	5	CIRCLE MILL .750 HOLE	Circle Mill	0	-0.53	00:01:50	Flood	PC, left	5	4500	abs -0.17
	Spot Drill 1/2 SPOTDRILL	3	SPOT DRILL X12 TOP OF BOSS	Simple drill - no peck	0	-0.1	00:00:00			3	2000	abs -0.03
	Drill NO. 36 DRILL	4	SPOT DRILL X12 TOP OF BOSS	Peck drill - full retract	0	-0.56	00:00:00	Flood		4	4800	abs -0.03

Post the G Code to the post processor

Tip: *Learn how to read code and check your work!*



```
O0010 (PIVOTCLAMP_LAB)
( DATE=DD-MM-YY - 28-06-12 TIME=HH:MM - 13:12 )
(MCX FILE - C:\DOCUMENTS AND SETTINGS\KOENIG\MY DOCUMENTS\CAE_7928\FINAL_HANDOUT\PIVOTCLAMP_LAB.MCX)
(NC FILE - C:\DOCUMENTS AND SETTINGS\KOENIG\MY DOCUMENTS\CAE_7928\FINAL_HANDOUT\PIVOTCLAMP_LAB.NC)
(MATERIAL - ALUMINUM INCH - 6061)
( T1 | 2" FACE MILL | H1 )
( T2 | 3/4 FLAT ENDMILL | H2 )
( T5 | 1/4 FLAT ENDMILL | H5 )
( T3 | 1/2 SPOTDRILL | H3 )
( T4 | NO. 36 DRILL | H4 )
( T9 | 1/2 FLAT ENDMILL | H9 )
N100 G20
N102 G0 G17 G40 G49 G80 G90
( FACE TOP OF STOCK )
N104 T1 M6
N106 G0 G90 G54 X-1.35 Y-1. A0. S1200 M3
N108 G43 H1 Z2. M8
N110 Z0.
N112 G1 Z-.025 F5.
N114 X5.55 F10.
N116 G0 Z2.
N118 M5
N120 G91 G28 Z0. M9
N122 A0.
N124 M01
( MILL TOP FACE FOR BOSS )
N126 T2 M6
N128 G0 G90 G54 X1.9256 Y.9236 A0. S1200 M3
N130 G43 H2 Z2. M8
N132 Z.1
N134 G1 Z-.15 F5.
```

Ready... CAPE Line: 2 Col: 21 File Size: 14 kb 6/28/2012 1:12 PM

G Codes Preparatory Functions

HAAS C.N.C. MILL PREPARATORY FUNCTIONS

MODAL
NON-MODAL
DEFAULT *
OPTIONAL **

G00* RAPID POSITIONING MOTION (X,Y,Z,A,B) (SETTING 10, 56, 101)	G73 HIGH SPEED PECK DRILL CANNED CYCLE (X,Y,A,B,Z,I,J,K,Q,P,R,L,F) (SETTING 22, 52)
G01 LINEAR INTERPOLATION MOTION (X,Y,Z,A,B,F)	G74 REVERSE TAPPING CANNED CYCLE (X,Y,A,B,Z,I,J,K,Q,P,R,L,F) (SETTING 130, 133)
G02 CIRCULAR INTERPOLATION MOTION CW (X,Y,Z,A,I,J,K,R,F)	G76 FINE BORING CANNED CYCLE (X,Y,A,B,Z,I,J,P,Q,R,L,F) (SETTING 27)
G03 CIRCULAR INTERPOLATION MOTION CCW (X,Y,Z,A,I,J,K,R,F)	G77 BACK BORE CANNED CYCLE (X,Y,A,B,Z,I,J,Q,R,L,F) (SETTING 27)
G04 DWELL (P) (P=seconds", *milliseconds)	G80* CANCEL CANNED CYCLE (SETTING 56)
G09 EXACT STOP, NON-MODAL	G81 DRILL CANNED CYCLE (X,Y,A,B,Z,R,L,F)
G10 PROGRAMMABLE OFFSET SETTING (X,Y,Z,A,L,P,R)	G82 SPOT DRILL / COUNTERBORE CANNED CYCLE (X,Y,A,B,Z,P,R,L,F)
G12 CW CIRCULAR POCKET MILLING (Z,I,K,Q,D,L,F)	G83 PECK DRILL DEEP HOLE CANNED CYCLE (X,Y,A,B,Z,I,J,K,Q,P,R,L,F) (SETTING 22, 52)
G13 CCW CIRCULAR POCKET MILLING (Z,I,K,Q,D,L,F)	G84 TAPPING CANNED CYCLE (X,Y,A,B,Z,I,J,R,L,F) (SETTING 130, 133)
G17* CIRCULAR MOTION XY PLANE SELECTION (G02 or G03) (SETTING 56)	G85 BORE IN-BORE OUT CANNED CYCLE (X,Y,A,B,Z,R,L,F)
G18 CIRCULAR MOTION ZX PLANE SELECTION (G02 or G03)	G86 BORE IN-STOP-RAPID OUT CANNED CYCLE (X,Y,A,B,Z,R,L,F)
G19 CIRCULAR MOTION YZ PLANE SELECTION (G02 or G03)	G87 BORE IN-MANUAL RETRACT CANNED CYCLE (X,Y,A,B,Z,R,L,F)
G20* VERIFY INCH COORDINATE POSITIONING (SETTING 9, set to INCH)	G88 BORE IN-DWELL-MANUAL RETRACT CANNED CYCLE (X,Y,A,B,Z,P,R,L,F)
G21 VERIFY METRIC COORDINATE POSITIONING (SETTING 9 set to METRIC)	G89 BORE IN-DWELL-BORE OUT (X,Y,A,B,Z,P,R,L,F)
G28 MACHINE ZERO RETURN THRU REF. POINT (X,Y,Z,A,B) (SETTING 108)	G90* ABSOLUTE POSITIONING COMMAND
G29 MOVE TO LOCATION THROUGH G28 REF. POINT (X,Y,Z,A,B)	G91 INCREMENTAL POSITIONING COMMAND (SETTING 29)
G31** FEED UNTIL SKIP FUNCTION (X,Y,Z,A,B,F)	G92 GLOBAL WORK COORDINATE SYSTEM SHIFT (FANUC) (HAAS) (SETTING 33)
G35** AUTOMATIC TOOL DIAMETER MEASUREMENT (D,H,Z,F)	G92 SET WORK COORDINATE VALUE (YASNAC) (SETTING 33)
G36** AUTOMATIC WORK OFFSET MEASUREMENT (X,Y,Z,A,B,I,J,K,F)	G93 INVERSE TIME FEED MODE ON
G37** AUTOMATIC TOOL LENGTH MEASUREMENT (D,H,Z,F)	G94* INVERSE TIME FEED MODE OFF / FEED PER MINUTE ON (SETTING 56)
G40* CUTTER COMP CANCEL G41/G42/G141 (X,Y)	G95 FEED PER REVOLUTION (SETTING 9, 56)
G41 2D CUTTER COMPENSATION, LEFT (X,Y,D) (SETTING 40, 43, 44, 58)	G98* CANNED CYCLE INITIAL POINT RETURN (SETTING 56)
G42 2D CUTTER COMPENSATION, RIGHT (X,Y,D) (SETTING 40, 43, 44, 58)	G99 CANNED CYCLE "R" PLANE RETURN
G43 TOOL LENGTH COMPENSATION+ (H,Z) (SETTING 15)	G100 MIRROR IMAGE G101 CANCEL
G44 TOOL LENGTH COMPENSATION- (H,Z) (SETTING 15)	G101 MIRROR IMAGE (X,Y,Z,A,B) (SETTING 45, 46, 47, 48, 80)
G47 TEXT ENGRAVING (X,Y,Z,R,I,J,P,E,F) (Macro Variable 599 Change Serial #)	G102 PROGRAMMABLE OUTPUT TO RS-232 (X,Y,Z,A,B)
G49* TOOL LENGTH COMPENSATION CANCEL G43/G44/G143 (SETTING 56)	G103 LIMIT BLOCK LOOKAHEAD (P0-P15 for number control looks ahead)
G50* SCALING G51 CANCEL (SETTING 56)	G107 CYLINDRICAL MAPPING (X,Y,Z,A,Q,R)
G51** SCALING (X,Y,Z,P) (SETTING 71)	G110-G129 WORK OFFSET POSITIONING COORDINATE #7-26
G52 WORK OFFSET POSITIONING COORDINATE (SETTING 33, YASNAC)	G136** AUTOMATIC WORK OFFSET CENTER MEASUREMENT
G52 GLOBAL WORK COORDINATE OFFSET SHIFT (SETTING 33, FANUC)	G141 3D+ CUTTER COMPENSATION (X,Y,Z,I,J,K,D,F)
G52 GLOBAL WORK COORDINATE OFFSET SHIFT (SETTING 33, HAAS)	G143** 5-AXIS TOOL LENGTH COMPENSATION+ (X,Y,Z,A,B,H) (SETTING 15, 117)
G53 MACHINE ZERO XYZ POSITIONING, NON-MODAL (X,Y,Z,A,B)	G150 GENERAL PURPOSE POCKET MILLING (X,Y,P,Z,I,J,K,Q,D,R,L,S,F)
G54* WORK OFFSET POSITIONING COORDINATE #1 (SETTING 56)	G153** 5-AXIS HIGH SPEED PECK DRILL CANNED CYCLE (X,Y,A,B,Z,I,J,K,Q,P,E,L,F) (SETTING 22)
G55 WORK OFFSET POSITIONING COORDINATE #2	G154** SELECT WORK OFFSET POSITIONING COORDINATE P1-99 (P)
G56 WORK OFFSET POSITIONING COORDINATE #3	G155** 5-AXIS REVERSE TAPPING CANNED CYCLE (X,Y,A,B,Z,I,J,E,L,F)
G57 WORK OFFSET POSITIONING COORDINATE #4	G161** 5-AXIS DRILL CANNED CYCLE (X,Y,A,B,Z,E,L,F)
G58 WORK OFFSET POSITIONING COORDINATE #5	G162** 5-AXIS SPOT DRILL / COUNTERBORE CANNED CYCLE (X,Y,A,B,Z,P,E,L,F)
G59 WORK OFFSET POSITIONING COORDINATE #6	G163** 5-AXIS PECK DRILL CANNED CYCLE (X,Y,A,B,Z,I,J,K,Q,P,E,L,F) (SETTING 22)
G60 UNI-DIRECTIONAL POSITIONING (X,Y,Z,A,B) (SETTING 35)	G164** 5-AXIS TAPPING CANNED CYCLE (X,Y,A,B,Z,I,E,L,F)
G61 EXACT STOP, MODAL (X,Y,Z,A,B)	G165** 5-AXIS BORE IN, BORE OUT CANNED CYCLE (X,Y,A,B,Z,E,L,F)
G64* EXACT STOP G61 CANCEL (SETTING 56)	G166** 5-AXIS BORE IN, STOP, RAPID OUT CANNED CYCLE (X,Y,A,B,Z,E,L,F)
G65** MACRO SUB-ROUTINE CALL	G169** 5-AXIS BORE IN, DWELL, BORE OUT (X,Y,A,B,Z,P,E,L,F)
G68** ROTATION (G17,G18,G19,X,Y,Z,R) (OPTION) (SETTING 72, 73)	G174 NON-VERTICAL RIGID TAPPING CCW (X,Y,Z,F)
G69* ROTATION G68 CANCEL (SETTING 56)	G184 NON-VERTICAL RIGID TAPPING CW (X,Y,Z,F)
G70 BOLT HOLE CIRCLE with a CANNED CYCLE (I,J,L)	G187 ACCURACY CONTROL FOR HIGH SPEED MACHINING (E) (SETTING 85)
G71 BOLT HOLE ARC with a CANNED CYCLE (I,J,K,L)	G188 G188 GET PROGRAM FROM PST (Program Schedule Table)
G72 BOLT HOLES ALONG AN ANGLE with a CANNED CYCLE (I,J,L)	

M Codes – Misc. Functions

HAAS C.N.C. MILL MISCELLANEOUS FUNCTIONS

OPTION**

M00	PROGRAM STOP (SETTING 39, 42)	M50**	EXECUTE PALLET CHANGE (P) (SETTING 121 thru 129)
M01	OPTIONAL PROGRAM STOP (SETTING 17, 39)	M51-M58	OPTIONAL USER M CODE SET
M02	PROGRAM END (SETTING 39)	M59	OUTPUT RELAY SET (N)
M03	SPINDLE ON CLOCKWISE (S) (SETTING 144)	M61-M68	OPTIONAL USER M CODE CLEAR
M04	SPINDLE ON COUNTERCLOCKWISE (S) (SETTING 144)	M69	OUTPUT RELAY CLEAR (N)
M05	SPINDLE STOP	M75	SET G35 OR G136 REFERENCE POINT
M06	TOOL CHANGE (T) (SETTING 42, 87, 155)	M76	CONTROL DISPLAY INACTIVE
M08	COOLANT ON (SETTING 32)	M77	CONTROL DISPLAY ACTIVE
M09	COOLANT OFF	M78	ALARM IF SKIP SIGNAL FOUND
M10**	4th AXIS BRAKE ON	M79	ALARM IF SKIP SIGNAL NOT FOUND
M11**	4th AXIS BRAKE RELEASE	M80**	AUTOMATIC DOOR OPEN (SETTING 131)
M12**	5th AXIS BRAKE ON	M81**	AUTOMATIC DOOR CLOSE (SETTING 131)
M13**	5th AXIS BRAKE RELEASE	M82	TOOL UNCLAMP
M16	TOOL CHANGE (T) (same as M06)	M83**	AUTO AIR JET ON
M17**	APC PALLET UNCLAMP and OPEN APC DOOR	M84**	AUTO AIR JET OFF
M18**	APC PALLET CLAMP and CLOSE DOOR	M86	TOOL CLAMP
M19	ORIENT SPINDLE (P,R values optional)	M88**	COOLANT THROUGH SPINDLE ON (SETTING 32)
M21-M28	OPTIONAL USER M CODE INTERFACE WITH M-FIN SIGNAL	M89**	COOLANT THROUGH SPINDLE OFF (SETTING 32)
M30	PROGRAM END AND RESET (SETTING 2, 39, 56, 83)	M95	SLEEP MODE
M31	CHIP AUGER FORWARD (SETTING 114, 115)	M96	JUMP IF NO INPUT (P,Q)
M33	CHIP AUGER STOP	M97	LOCAL SUB-PROGRAM CALL (P,L)
M34	COOLANT SPIGOT POSITION DOWN, INCREMENT (+1)	M98	SUB-PROGRAM CALL (P,L)
M35	COOLANT SPIGOT POSITION UP, DECREMENT (-1)	M99	SUB/LOCALSUB-PROGRAM / RETURN OR LOOP (P) (SETTING 118)
M36**	PALLET PART READY (P)	M101	MOM (Minimum Oil Machining) CANNED CYCLE MODE (I)
M39	ROTATE TOOL TURRET (T#) (SETTING 86)	M102	MOM MODE (I,J)
M41	SPINDLE LOW GEAR OVERRIDE	M103	MOM MODE CANCEL
M42	SPINDLE HIGH GEAR OVERRIDE	M109**	INTERACTIVE USER INPUT (P)

All M codes are effective or cause an action at the end of the block and only one M code is allowed in each block.

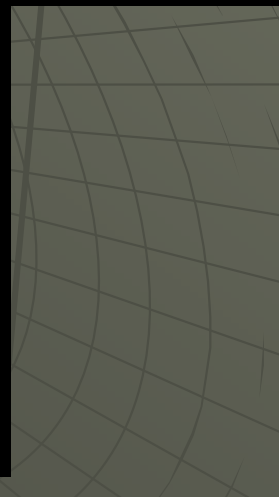
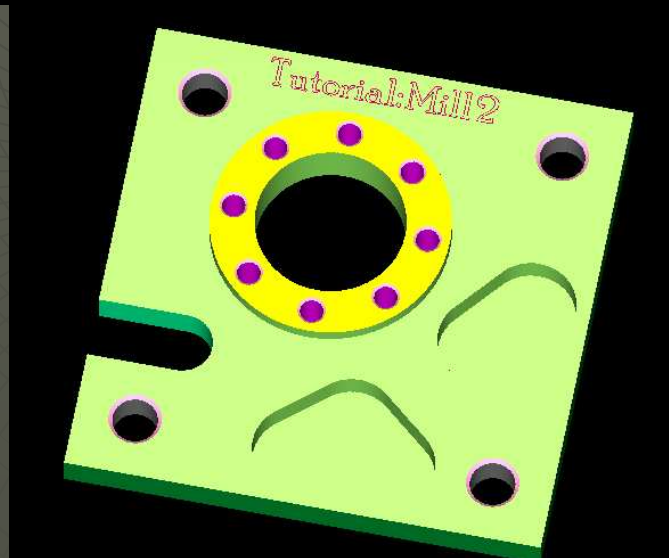
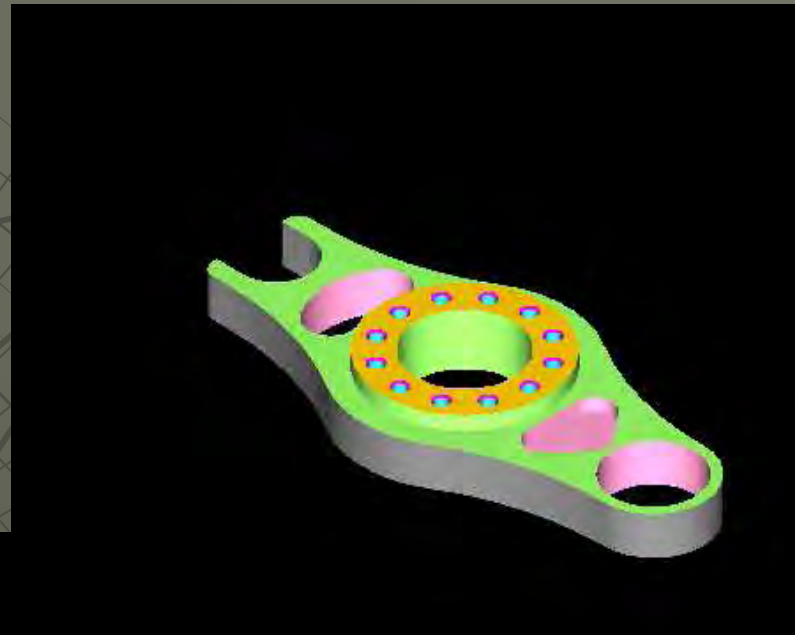
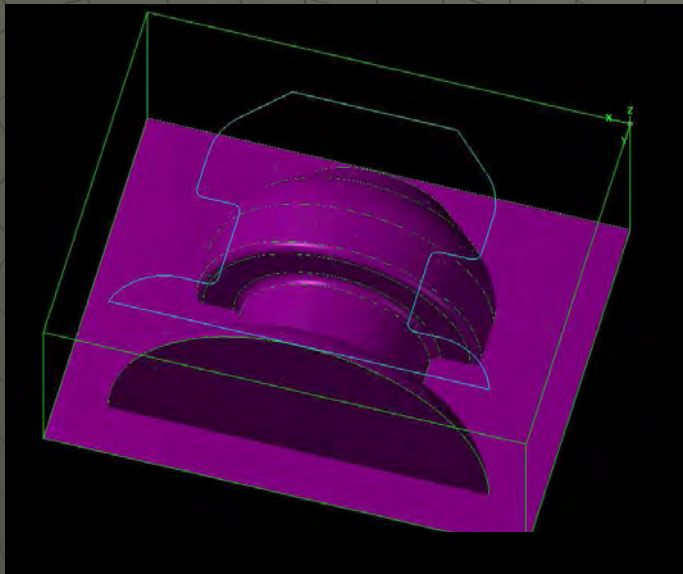


Updated on 1-01-06

Project Report Requirements

- ◆ Cover page
- ◆ SolidWorks detail drawing: Pivot Clamp
- ◆ Print all toolpaths page Mastercam
- ◆ Print tool list page Mastercam
- ◆ Print set up sheet page Mastercam
- ◆ Print the 1st page of the NC Code & Annotate the major G & M code operations on this page – see G & M Code sheets provided)

Some parts modeled in SolidWorks (CAD) toolpaths created using Mastercam (CAM)...&?



Oh...one last thing!

When opening Mastercam you will need to:

Select:

- ◆ Start
- ◆ All Programs
- ◆ Mastercam X4
- ◆ Nhasp X
- ◆ Select NetHASP button
- ◆ Read
- ◆ Click Activate Licenses button
- ◆ OK